Natural Sciences and Engineering Research Council of Canada

Québec
Ministère des affaires internationales

USSR Academy of Sciences

ALCAN

UNITED TECHNOLOGIES PRATT & WHITNEY CANADA

BOMBARDIER canadair

IBM

sun microsystems

SECOND NORTH AMERICAN-SOVIET WORKSHOP ON COMPUTATIONAL AERODYNAMICS

– ABSTRACTS –

CFD Lab
Concordia University

Concordia
UNIVERSITY

CRAY RESEARCH, INC.

ÉCOLE POLYTECHNIQUE DE MONTRÉAL

CONVEX

FPS COMPUTING

McGill University

SiliconGraphics
Computer Systems

Bell Helicopter TEXTRON

BOEING
Boeing of Canada Ltd.
de Havilland Division

CONTROL DATA

Cet atelier est dédié à la mémoire des 14
ingénieures et étudiantes de
l'École Polytechnique de Montréal
dont la vie fut si soudainement et
futilement abrégée
et
à la mémoire de l'académicien
Boris Grigorievich Galerkin
à qui la science doit énormément

This Workshop is dedicated to the
memory of the
14 women engineers and students
of the
École Polytechnique de Montréal
whose lives were
so suddenly and futilely wasted
and
to the memory of Academician
Boris Grigorievich Galerkin
to whom science owes a great debt of
gratitude

2nd North American-Soviet Workshop on Computational Aerodynamics

PREFACE

In October 1989, in the spirit of Glasnost and Perestroïka, a group of American Aerospace scientists were invited by the USSR Academy of Sciences to participate in a joint Soviet-American Workshop on Computational Aerodynamics. This open encounter, initiated by Academician Oleg Belotserkovsky and coordinated in the USA by Professor Walter L. Melnik of Maryland, was the first of its kind in the field of aerospace. The American scientists invited included industrial leaders, academics and US Agencies representatives such as the various NASAs. A delay in signing an official scientific exchange agreement between the American and Soviet Governments meant, however, that NASA scientists did not join the team. I had then the privilege to be asked to join the American Delegation on that trip. The group landed in Moscow and from there flew to Tashkent, Uzbekistan, where the Workshop took place. Following the Workshop, several members of the team were able to visit the major aerospace research installations at Leningrad, Novosibirsk and other places.

The meeting was a friendly and informative encounter, and showed that Soviet scientists compensated for a relative lack of computer power with innovative use of mathematics. Both sides were pleased with the exchange and promised, in an informal signed document, to attempt to nurture the relationship by:

1. adopting a proposal by Professor Mohamed Hafez to initiate a CFD Journal to bridge the East-West scientific and language barriers. Professor Hafez and Academician Belotserkovsky were mandated to organize this Journal. Other co-signers were expected to become members of the Editorial Board.

2. holding a bi-annual Workshop.

It was decided that the next Workshop would be held in Montreal and would be renamed North American-Soviet Workshop on Computational Aerodynamics (SNA-WCA), to reflect a broader Canadian-US participation. In October '90 Concordia University applied for, and was awarded, a Conference grant from the Natural Sciences and Engineering Research Council of Canada (NSERC). To make the Conference self-financing other sources of funding were solicited from industry, universities and the Provincial government. Despite the ominous recession of 1991, the response was excellent and the list of sponsors is indeed an impressive one.

This rapprochement in the aerospace and computer fields is important and this Workshop is a good opportunity for Canadian industry and agencies to be part of the inevitable scientific and economic exchange process between East and West.

The Workshop should offer the opportunity to size up the Soviet advances and contrast the applied and research aerospace technologies of East and West, at the highest levels. The opportunity to meet such a number of Soviet scientists, affiliated with the USSR Academy of Sciences, outside of the Soviet Union, is indeed rare.

In addition, a German and a Japanese representative will complement this by presenting their countries' state-of-the-art in aerodynamics.

I hope that you will appreciate the timeliness of this Workshop and that you will find it to be an important and useful interaction.

Finally, I would to extend my greatest thanks to Dr. Leonid Turchak, my co-organizer, who was tireless in his efforts to make all the Soviet arrangements.

W.G. Habashi
Department of Mechanical Engineering
Concordia University
Montreal, September 3, 1991

SECOND NORTH AMERICAN-SOVIET WORKSHOP

ON

COMPUTATIONAL AERODYNAMICS

Concordia University, Montreal, Canada

September 3-5, 1991

PROGRAMME

TUESDAY SEPTEMBER 3

Opening Remarks, 8:30 am - 9:00 am

Dr. Peter Morand
President, Natural Sciences and Engineering Research Council of Canada (NSERC)

Dr. M.N.S. Swamy
Dean, Faculty of Engineering and Computer Science, Concordia University

Dr. Wagdi G. Habashi
Conference Chairman

OVERVIEW OF WORKSHOP PROGRAMME AND ACTIVITIES

	Morning 8:30 am - 11:45 am	11:45 am - 1:00 pm	Afternoon 1:00 pm - on	Evening
Tuesday September 3	Soviet Sessions 1 and 2	Lunch Break	North American Sessions 1 and 2	Reception 5:30 - 7:30 pm (tickets)
Wednesday September 4	Soviet Sessions 3 and 4	Lunch Break	North American Sessions 3 and 4 Euro-Asian Session	Banquet 6:30 for 7:00 pm (tickets)
Thursday September 5	Soviet Sessions 5 and 6	Lunch Break	North American Sessions 5 and 6	

TUESDAY SEPTEMBER 3

SOVIET SESSION 1

Chairman: M.M. Hafez, University of California

9:00- 9:30	**Sergei K. Godunov** USSR Academy of Sciences Mathematical Institute, Novosibirsk *Liapunov Functions for Flutter Problems*
9:30 - 10:00	**Alexei A. Kolganov** Zhukovsky Central Aero-Hydrodynamical Institute *The Method of Discrete Vortices for Industrial Aerodynamics*

10:00 - 10:15 Coffee Break

SOVIET SESSION 2

Chairman: W. Schmidt, MBB-Deutsche Aerospace

10:15 - 10:45	**V.P. Korobeinikov** USSR Academy of Sciences FEB Institute of Applied Mathematics, Vladivostok *On Modeling of Processes in Pulse Jet Engines*
10:45 - 11:15	**I.N. Murzinov** Central Research Institute of Machine-Building, Moscow *Heat Transfer and Protection Problems for the Energia-Buran (Space Shuttle) System*
11:15 - 11:45	**V.Y. Neyland** Zhukovsky Central Aero-Hydrodynamical Institute, Moscow *Aero-Thermodynamics of Space Planes*

11:45 - 1:00 Lunch Break

TUESDAY SEPTEMBER 3

NORTH AMERICAN SESSION 1

CFD Perspective - The Aerospace Industry
Chairman: B. Eggleston, Boeing of Canada - de Havilland Division

1:00 - 1:30 **Fotis D. Mavriplis**
Manager, Advanced Aerodynamics, Canadair Aerospace Group, Bombardier Inc., Canada
CFD Developments and Applications at Canadair

1:30 - 2:00 **Paul E. Rubbert**
Unit Chief, Aerodynamics Research, Boeing Commercial Airplane, USA
The Role of CFD in Airplane Design

2:00 - 2:30 **Ramesh K. Agarwal**
Director CFD, McDonnell Douglas Research Laboratories, USA
Recent Advances in Computational Aerodynamics

2:30 - 3:00 **Vijaya Shankar**
Manager CFD, Rockwell International Corporation, Science Center, USA
Impact of Supercomputing in the Aerospace Sciences

3:00 - 3:15 Coffee Break

NORTH AMERICAN SESSION 2

Finite Element and Control Volume Techniques - I
Chairman: R. Camarero, École Polytechnique de Montréal

3:15 - 3:45 **B. Rabi Baliga**
Professor, Department of Mechanical Engineering, McGill University, Canada
Co-located Equal-Order Primitive Variables Formulations in CVFEM

3:45 - 4:15 **Roy A. Nicolaides**
Professor, Department of Mathematics, Carnegie-Mellon University, USA
Complementary Volume Schemes in Incompressible Flow

4:15 - 4:45 **Michel Fortin**
Professor, Department of Mathematics and Statistics, Université Laval, Canada
A Posteriori Error Estimators in Compressible Viscous Flows Simulation

4:45 - 5:15 **Gerry Schneider**
Professor, Department of Mechanical Engineering, University of Waterloo, Canada
Advances in Control Volume Based Finite Element Methods for Compressible Flows

5:30 pm Reception, Hotel Europa (tickets)

WEDNESDAY SEPTEMBER 4

SOVIET SESSION 3

Chairman: Academician O. Belotserkovsky
USSR Academy of Sciences Institute for Computer Aided Design

8:30 - 9:00 **Alexander S. Kholodov**
Physico-Technical Institute, Moscow
Grid-Characteristic Methods for Continuum Mechanics

9:00 - 9:30 **V.V. Shchennikov**
USSR Academy of Sciences Institute for Computer Aided Design, Moscow
New Efficient Numerical Methods for Gas Dynamics

9:30 - 10:00 **A.I. Tolstykh**
USSR Academy of Sciences Computing Centre, Moscow
High Accuracy Compact Upwind Schemes for Viscous Flows

10:00 - 10:15 Coffee Break

SOVIET SESSION 4

Chairman: Academician S. K. Godunov
USSR Academy of Sciences Mathematical Institute

10:15 - 10:45 **V.I. Shokin**
USSR Academy of Sciences Institute of Computational Technologies, Novosibirsk
Qualitative Analysis of Finite Difference Algorithms for Aerodynamics

10:45 - 11:15 **V.M. Kovenya**
USSR Academy of Sciences Institute of Theoretical and Applied Mechanics, Novosibirsk
Splitting Methods for Supersonic Flow Around Bodies

11:15 - 11:45 **E.G. Shifrin**
Association of Computational Fluid Dynamics, Moscow
A Numerical Hodograph Method for Aerodynamic Design

11:45 - 1:00 Lunch Break

WEDNESDAY SEPTEMBER 4

NORTH AMERICAN SESSION 3

CFD Applications - 1
Chairman: L. Turchak, USSR Academy of Sciences Computing Centre, Moscow

1:00 - 1:30	**Jim C. Narramore** Manager CFD, Bell Helicopter Textron, USA *Application of Navier-Stokes Methods to Rotorcraft Aerodynamic Design and Analysis*
1:30 - 2:00	**Mark Barnett** Senior Engineer, Computational Science Dept, United Technology Research Center, USA *Viscous-Inviscid Interaction Analysis of Compressor Cascade Performance*
2:00 - 2:30	**Andrew Pollard** Professor, Department of Mechanical Engineering, Queen's University, Canada *Modeling of Drag Reduction by Passive Means*

2:30- 2:45 Coffee Break

NORTH AMERICAN SESSION 2

Finite Element and Control Volume Techniques - III
Chairman: R.A. Nicolaides, Carnegie-Mellon University

2:45 - 3:15	**Klaus-Jürgen Bathe** Professor, Department of Mechanical Engineering, Massachusetts Institute of Technology, USA *On the Finite Element Solution of Incompressible Fluid Flows with Heat Transfer*
3:15 - 3:45	**J.N. Reddy** Professor, Department of Engineering Science & Mechanics, Virginia Polytechnic Institute & State University, USA *Element-by-Element Methods for the Solution of Incompressible Three-Dimensional Flows*
3:45 - 4:15	**Wagdi G. Habashi** Professor, Department of Mechanical Engineering, Concordia University, Canada *A Finite Element Navier-Stokes Algorithm, Using Fully-Coupled Direct and Iterative Solvers*

4:15 - 4:30 Coffee Break

WEDNESDAY SEPTEMBER 4

THE EURO-ASIAN SESSION

CFD Perspective - The Aerospace Industry Overseas
Chairman: W.G. Habashi, Concordia University

4:30 - 5:00	**Wolfgang Schmidt** Director Air Vehicle Engineering, MBB-Deutsche Aerospace, Germany *Computational Aerodynamics in Germany*
5:00 - 5:30	**Koichi Oshima** Professor, Japan Society of Computational Fluid Dynamics *Recent CFD Research and Development in Asia-West Pacific Area*

6:30 pm Cocktails for 7:00 pm Banquet, Hotel Europa (tickets)

THURSDAY SEPTEMBER 5

SOVIET SESSION 5

Chairman: V.V. Shchennikov, USSR Academy of Sciences Computing Centre, Moscow

8:30 - 9:00 **A.V. Babakov**
Association of Computational Fluid Dynamics, Moscow
Simulation of Subsonic and Transonic Unsteady Flows

9:00 - 9:30 **Y.P. Golovachev**
IOFFE Physico-Technical Institute, Leningrad
Numerical Modeling of Supersonic Unsteady Flows for Bodies in a Non-Homogeneous Medium

9:30 - 10:00 **Leonid I. Turchak**
USSR Academy of Sciences Computing Centre, Moscow
Numerical Simulation of Unsteady and Non-Equilibrium Supersonic Flows

10:00- 10:15 Coffee Break

SOVIET SESSION 6

Chairman: W.H. Finlay, University of Alberta

10:15 - 10:45 **Oleg M. Belotsertkovsky**
Director, USSR Academy of Sciences Institute for Computer Aided Design, Moscow
Direct Numerical Simulation in Turbulence

10:45 - 11:15 **V.S. Ryaben'kii**
USSR Academy of Sciences Keldish Institute of Applied Mathematics, Moscow
On the Difference Potential Method for CFD

11:15 - 11:45 **V.I. Gushchin**
Scientific Director, USSR Academy of Sciences Institute for Computer Aided Design, Moscow
Direct Numerical Simulation of Transitional Separated Flows

11:45- 1:00 Lunch Break

6 30 pm
1440 Ste. Catherine
424

THURSDAY SEPTEMBER 5

NORTH AMERICAN SESSION 5

Techniques for Higher Accuracy
Chairman: M.D. Gunzburger, Virginia Polytechnic Institute & State University

1:00- 1:30 **Ami Harten**
Professor, Department of Mathematics, University of California, Los Angeles, USA
Multi-Dimensional Essentially Non-Oscillatory (ENO) Schemes for General Geometries

1:30 - 2:00 **Mohamed M. Hafez**
Professor, Department of Mechanical Engineering, University of California, Davis, USA
Control of Artificial Viscosity Effects in Flow Simulationss

2:00 - 2:30 **Ricardo Camarero**
Professor, Department of Mechanical Engineering, École Polytechnique de Montréal, Canada
Adaptive and Moving Meshes

2:30 - 3:00 **Warren H. Finlay**
Professor, Department of Mechanical Engineering, University of Alberta, Canada
A Spectral Element Method Applied to Parabolized Simulations of Stability and Transition

3:00 - 3:15 Coffee Break

NORTH AMERICAN SESSION 6

CFD Applications - II
Chairman: G. Schneider, University of Waterloo

3:15 - 3:45 **Max D. Gunzburger**
Professor, Department of Mathematics, Virginia Polytechnic Institute & State University, USA
Algorithms for Flow Optimization and Control

3:45 - 4:15 **Walter Melnik**
Professor, Department of Aerospace Engineering, University of Maryland, USA
Non-equilibrium Viscous Shock-Layer Calculations of Glassy Ablators

4:15 - 4:45 **Grafton W. H. Hui**
Professor, Department of Applied Mathematics, University of Waterloo, Canada
A New Lagrangian Approach for Solving the Euler Equations

4:45- 5:30

Panel Discussion (Coffee)

The Future of Computational Aerodynamics-East and West
Possible Collaborations

5:30 pm: END

Soviet Abstracts

Simulation of Subsonic and Transonic Unsteady Flows

A.V. Babakov

USSR Academy of Sciences Institute of Computer Aided Design
2nd Brestskaya 19/18, Moscow 123865, USSR

ABSTRACT

The numerical simulation of subsonic and transonic flows is being carried out on the basis of conservative difference laws. The Euler, Navier-Stokes and the semi-empirical k-ε turbulence model are used to describe the medium. The numerical simulation of the unsteady near-wake past a cylinder is being carried out based on these models.

The method developed allows useage of effective parallel algorithms for two- and three-dimensional flow simulations. For practical aerodynamic problems, the speed-up characteristics of the algorithm are demonstrated on an increasing number of processors. The efficiency of such parallel algorithms is illustrated by the simulation of three-dimensional flow over a car.

Direct Numerical Simulation in Turbulence

Academician Oleg M. Belotserkovsky

Director, USSR Academy of Sciences Institute of Computer Aided Design
2nd Brestskaya Str. 19/18, 123056 Moscow

ABSTRACT

Direct numerical modelling of the complete unsteady dynamical equations is carried out by means of a method of splitting over physical processes. The problem under investigation is that of the evolution of coherent structures for the limiting regime (Reynolds number=∞) of the free shear turbulence in the near-wake behind a body moving in a fluid, as well as the process of transition of laminar flow into irregular state (chaos), by the study of oceanic vortical structures, emerging under the influence of wind-induced stresses. A discrete dissipation model, obtained from the non-stationary Euler equations (the conservation laws) by averaging over an elementary cell and time interval, is proposed for the modelling of "ordered" motions and large-scale vortices. The stochastic component of turbulence (background) is modelled at the kinetic level by a statistical Monte Carlo method. Such an approach sharply reduces the level of demand on computer resources. The efficiency of the algorithm proposed is demonstrated by the solution of the evolution of the turbulent wake for two- and three-dimensional flows in the supercritical and transonic regimes.

The investigation of the vortices production in oceanic flow is conducted on the basis of nonlinear equations taking into account a turbulent dissipation and the earth's rotation (β-effect, connected with a latitudinal alteration of Coriolis parameter), by the presence of wind-induced stresses. Here, at the highest derivatives appear two small parameters, which characterize the effects of the earth's rotation $\varepsilon = (\{v/\beta\}^{1/2}/L)$ and of viscosity ($\gamma=\varepsilon/Re^{1/3}$). This leads to the formation at the western boundary of an inertial-viscous boundary layer. With increasing Reynolds number, and at small values of $\varepsilon \cong 0.02$, one obtains from the calculations the following succession of flow regimes:

(1) stationary ($Re \cong 0.25$),
(2) periodical ($Re \cong 1\text{-}2$),
(3) stable irregular ($Re \cong 2.5$), and
(4) unstable irregular ($Re \cong 3$).

The last of these regimes is characterized by a sharp growth of the kinetic energy (a numerical study of "Kolmogoroff flow" failed to obtain the transition to chaos). When using medium-power computers, the models listed above permit to obtain results agreeing well with experimental data.

Liapunov Functions for Flutter Problems

Sergei K. Godunov

USSR Academy of Sciences
SB Institute of Mathematics, 630 090 Novosibirsk, USSR

ABSTRACT

The stability of the linear system ($\varepsilon \geq 0$, $\nu > 0$):

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{y}_i}\right) + \frac{\partial \Phi}{\partial y_i} + \varepsilon \frac{\partial R}{\partial \dot{y}_i} = \nu f_i$$

$$\Phi = \sum \Phi_{ij} y_i y_j \geq 0$$

$$T = \sum T_{ij} \dot{y}_i \dot{y}_j > 0$$

$$R = \sum R_{ij} \dot{y}_i \dot{y}_j > 0$$

shall be studied. Such equations describe the oscillations in aeroelasticity, if:

$$\nu f_i = \nu \sum f_{ik} y_k$$

are the aerodynamic forces. It is proposed that, if Φ=0, then R=0, f_i=0.

We will present a procedure for the Liapunov functions construction that uses the averaged energy integral and the linear algebraic algorithms only, for symmetrical matrices. Such algorithms yield a guaranteed accuracy estimate.

Numerical Modeling of Supersonic Unsteady Flows for Bodies in a Non-Homogeneous Medium

Y.P. Golovachev

Ioffe Physico-Technical Institute
Polytechnic Street 26, 194 021, Leningrad, USSR

ABSTRACT

A nonlinear formulation is applied to investigate the supersonic axisymmetric flows about blunt bodies, moving through domains with variable gas temperature, non-equilibrium internal energy of molecules, and containing solid particles. These unsteady flows are characterized by generation of steep flow gradients within the shock layer, in particular intense compression and rarefaction waves interacting with the bow shock and the body surface. Bearing in mind these features of the flows under study, numerical simulation is carried out within the framework of the viscous shock-layer approach, that allows the calculation of the heat transfer when conventional boundary layer theory fails, even for high Reynolds numbers.

The problem is solved on the basis of the reduced Navier Stokes equations, which include all terms of the inviscid flow and boundary layer equations. Numerical solutions are obtained using implicit finite-difference schemes. The computational domain is bounded by the bow shock, so the grid points are moving in accordance with the shock motion. Grid adaptation to the flow field structure is also applied at each time step, based on an equidistribution principle using a linear combination of flow gradients to govern grid points clustering.

Calculations have demonstrated significant non-stationary effects on the shock-layer flowfield structure, and on drag and heat transfer parameters variations along the body surface.

Direct Numerical Simulation of Transitional Separated Flows

V.A. Gushchin

USSR Academy of Sciences Institute of Computer Aided Design
2-nd Brestskaya, 19/18, Moscow, 123 865, USSR

ABSTRACT

The splitting method into physical processes is used for the calculations of the steady and unsteady incompressible flows past a circular cylinder, for a wide range of Reynolds numbers ($1<Re<10^6$). The finite-difference scheme for this method is second order accurate in space, has minimal viscosity and is monotonic.

While at moderate Reynolds numbers only alternative separation from both sides of the cylinder occurs, at large Reynolds numbers the unsteady flow is complicated by the existence of secondary effects such as the instability in separated shear layers during part of the period, the secondary vortices on the rear part of the cylinder surface and the secondary vortices near the reattachment point.

Our calculations, which are based on the modeling of large-scale vortex structures arising in unsteady separated flows around finite bodies, also allow the simulation of transitional fluid flows. The total drag coefficient crisis and the sharp rise in Strouhal number take place and are simulated numerically (without any turbulence models) at the critical Reynolds number ($Re \approx 4 \times 10^5$), with good agreement with experimental data and the calculations of other authors.

The numerical results confirm the suitability of the discrete model constructed to study separated fluid flow around finite bodies.

Grid-Characteristic Methods for Continuum Mechanics

A.S. Kholodov

Moscow Institute of Physics and Technology
141 700 Dolgoprudny, Moscow Region, USSR

ABSTRACT

A new approach, based on a linear analysis, is suggested for the construction of difference schemes, with a monotone approximation, on irregular grids and multiply connected domains. It is applied to the Laplace equation and to multi-dimensional transport equations. Its generalization to linear and nonlinear hyperbolic and elliptic systems is given.

Results of the numerical investigation of this type of schemes are given for steady and unsteady hydro- and aero-dynamic problems, for unsteady deformable rigid body problems and for biomechanics-type problems.

The Method of Discrete Vortices for Industrial Aerodynamics

Alexei A. Kolganov

Zhukovsky Central Aero-Hydrodynamical Institute
Radio Street 17, 107 005 Moscow, USSR

ABSTRACT

Fast methods of aerodynamic calculations are of great importance for problems of industrial aerodynamics, since at the preliminary stages costs should be kept to a reasonably level. A method that has proved highly efficient in achieving this goal is the method of discrete vortices (MDV), developed by Prof. S.M. Belotserkovsky. Its simplicity and apllicability to constructing diverse flow schemes, including separated and nonstationary flows, permits the solution of a large number of practical problems with sufficient accuracy.

The following problems are considered:

1. Jet problems of hydrodynamics:
   - outflow of stationary 3-D jets through nozzles of arbitrary configurations;
   - interaction of jets with obstacles;
   - the effect of water jets on ship rudders;
   - unsteady flow past lifting surfaces, including jet flows.

2. Evaluation of the aerodynamic characteristics of a car by means of two- and three-dimensional calculations.

3. Calculation of the unsteady aerodynamic characteristics of high-drag industrial structures, such as oil storage tanks.

Computed results are compared with experimental data and analytical solutions and the salient aerodynamic effects are discussed. The economics of the present approach are demonstrated.

On Modeling of Processes in Pulse Jet Engines

V.P. Korobeinikov

USSR Academy of Sciences FEB Institute of Applied Mathematics
Kirov Street 62, 690 068 Vladivostok

ABSTRACT

New schemes for developing engine thrust, based on mixed combustion, with additional heat deposition from electrodischarge, are studied. Suitable mathematical models are proposed which correspond to experiments in a combustion chamber with ignition and burning of mixtures of acetylene, hydrogen, methanol and other fuels with air. The motion of combustion products outlet tube are also considered.

The models are based on equations for quasi-one-dimensional unsteady flow of a gas mixture in a tube of variable cross-section. The combustion processes, wall friction, and heat losses are approximately taken into account.

The following problems are considered:

1. Local supply of mass and energy to the gases in tubes of variable cross-section. The spurted matter may be a gas or a two-phase mixture. The initial medium in the tubes and added mass can be either inert or combustible. Classes of self-similar problems have been qualitatively studied and numerical piecewise-continuous solutions of different types have been obtained.

2. Numerical modeling of the combustible gas flow in a combustion chamber and engine tubes is carried out. The nonlinear wave motion due to periodical combustion and heat addition by means of electric discharge (electro-chemical pulse engine) is considered. The Godunov and MacCormack finite-difference schemes and the method of integral relations were used for the numerical solution. The numerical model consists of two blocks. Performance of the combustion chamber for different burning regimes is modeled in the first block. The second block simulates space-time distribution of pressure, density and velocity in the system. The calculations are made for several initial data, system parameters and types of mixtures.

Estimations of impulse and thrust values are made and the effectiveness of the engine system is evaluated.

Splitting Methods for Supersonic Flow Around Bodies

V.M. Kovenya

USSR Academy of Sciences
SB Institute of Theoretical and Applied Mechanics
630 090 Novosibirsk

ABSTRACT

In the numerical simulation of aerodynamic problems, the models described by the complete and simplified Navier-Stokes equations for coercible heat-conduction gas are widely used. The use of the models' spectrum significantly reduces the cost of the simulation without sacrificing its details.

The present paper deals with the investigation of supersonic gas flows near bodies by means of the complete and simplified Navier-Stokes equations. The parabolic approximation of the Navier-Stokes equations was used as a simplified model. As a unified numerical method, the splitting technique into several physical processes and space directions was employed.

The first series of calculations deals with the influence of parameters of the incoming flow on the flow in the near-track behind the body including the case when the sting is present. The solution was carried out by means of the Navier-Stokes equations. The dependence of the points of break-away and adjunctions where obtained at different Mach and Reynolds numbers. It was shown that, as the Reynolds number increased, alongside with the primary vortex, a pair of vortex rings appeared in the vicinity of the break-away point which proves the hypothesis of vortex breakdown. It has been numerically shown that in the break-away area, local supersonic zones could exist.

In the second series of calculation, viscous flows were investigated for flows over space configurations, by means of Euler and simplified Navier-Stokes approximations. The behavior of aerodynamic characteristics with angle of attack were investigated.

Comparison of solutions for different models was carried out.

Heat Transfer and Protection Problems for the Energia-Buran Space Shuttle System

I.N. Murzinov

Central Research Institute of Machine-Building
Kaliningrad, Moscow Region, 141 070, USSR

ABSTRACT

A wide range of problems related to the aero-gasdynamics, heat transfer and thermal protection of multi-flight reusable space systems, is considered. The most complicated thermal problems are connected with the development of an orbiter. Its design and heat shield must ensure re-usability for hard re-entry conditions into the Earth's atmosphere.

The following problems are examined:

- choice of the orbiter's configuration, to ensure a permissible level of aerodynamic heating during re-entry;
- application of coatings with low catalytic activity for decreasing the temperature of the orbiter's surface;
- boundary layer transition and its influence on temperature distribution along the windward surface of the orbiter;
- heat expansion in complicated structures;
- heat transfer in splits between the tiles;
- thermal control of the orbiter outside the atmosphere.

Another set of thermal problems deals with the development of heavy launchers for orbital payloads. The launcher's engines result in high acoustic loads and intense heating of the launcher's base. The second peculiarity of atmospheric powered flight of multi-unit launchers is their composite aerodynamic configuration and formation of separation regions between the units. This results in the generation of local zones with extreme heating and pressure pulsations. Finally some problems arise in the development of thermal protection systems for propellant tanks using liquid hydrogen.

The paper includes results of numerical and experimental investigations, connected with the problems enumerated above.

Aero-Thermodynamics of Space Planes

V.Y. Neyland

Deputy Director, TsAGI
140 160 TsAGI, Zhukovsky, Moscow Region, USSR

ABSTRACT

The paper describes the role of numerical aero-thermodynamic methods in determining the configuration of space planes, in studying the thermal flows over them as they fly through the atmosphere, and in producing a pre-flight data bank of their aerodynamic characteristics.

As an example, the problem of the choice of a suitable mathematical model for the numerical study of the mission and the role of laboratory and/or flight experiments in mission approval are examined for the Energia-Buran Soviet Space Shuttle system.

Finally, the necessary developments in CFD techniques for the next generation of space planes (great cross-range in orbital descent; peculiarities of space plane with integrated configurations) are outlined.

On the Difference Potential Method for CFD

V.S. Ryaben'kii

USSR Academy of Sciences, Keldysh Institute of Applied Mathematics
125 047 Moscow, USSR

ABSTRACT

The Difference Potential Method (DPM) is intended for the numerical solution of linear differential equations, with variable coefficients, in arbitrary multidimensional domains, and having arbitrary boundary conditions (including nonlinear and nonlocal ones). The DPM can be treated as a technique for the discretization and numerical solution of the modified Boundary Equations with Projectors (BEP), proposed by A. Calderon. Formerly, Calderon's Boundary Equations have been used only for the qualitative investigation of the solutions.

Calderon's BEP and the DPM for the numerical solution of these equations have a wider domain of applicability in comparison with the classical potential theory, boundary equations and Boundary Element Method, as well as in comparison with the capacity matrix method. With the present method, there are no restrictions on the boundary conditions type, and knowledge of the fundamental solution is not necessary. Furthermore, unlike difference methods for the solution of boundary-value problems, difference approximation of boundary conditions is not required when using DPM.

In particular, DPM has been used for solving some inverse aerodynamic problems (D.S. Kamenetskii, E.G. Shifrin) and for artificial boundary conditions construction in stationary subsonic external flow problems solved by the Euler equations.

New Efficient Numerical Methods for Gas Dynamics

V.V. Shchennikov, Jr.

USSR Academy of Sciences Institute of Computer Aided Design
2nd Brestskaya Street 19/18, 123 056, Moscow, USSR

ABSTRACT

The work presented is an implementation of the discrete models ensemble approach (which corresponds to the concrete initial differential problem) for the modeling of the solution of the Euler equations, in contrast to the traditional approaches based on a single discrete model.

The idea of ensembling was first formulated by O.M. Belotserkovsky, A.I. Panarin and V.V. Shchennikov and suggested as the Rational Numerical Modelling (RNM) concept. The realization of the latter naturally assumes using computers with parallel architecture.

In the present work, within the framework of the RNM concept, we propose an original procedure which makes it possible to construct new methods of modelling of gas dynamics equations system solution (in Euler form). The main elements of the procedure are as follows:

- Euler equations system symmetrization,
- providing the property of asymptotic stability for each model in the ensemble,
- application of a preconditioning procedure to optimize each model of the ensemble.

The procedure of constructing the ensemble is invariant with respect to problem dimension.

Qualitative Analysis of Finite Difference Algorithms for Aerodynamics

Yu.I. Shokin

USSR Academy of Sciences SB Institute of Computational Technologies
630 090 Novosibirsk, USSR

ABSTRACT

This paper is concerned with the development of a numerical method for the prediction of 3-dimensional fluid flow with gravity waves in a bounded basin of finite depth. The method is based on a finite-difference discretization of the full potential, ideal, incompressible fluid flow model on a boundary-fitted curvilinear grid.

To illustrate the method's capabilities, the numerical solution of the problem of the inclined run-up of a solitary wave on a straight vertical wall is presented. The results are compared with experimental data and with a theoretical investigation of the problem.

A Numerical Hodograph Plane Method for Aeroodynamic Design

E.G. Shifrin

Association of Computational Fluid Dynamics
Vavilov Str. 40, 117 967, Moscow, USSR

ABSTRACT

The efficiency of an aerodynamic device is mostly defined by its form. Therefore, it is important to provide a general approach to the problem of designing the form to achieve a specified performance. The approach under consideration is based on the hodograph-plane method, where the profiling problem is formulated as a properly-posed boundary-value problem.

The problems considered are the following:

1. The profiling of the subsonic part of a Laval nozzle contour with straight sonic line and with a monotonic velocity distribution along the body. The supersonic part of the contour has a saddle point at the sonic line,

2. The profiling of the nozzle blades of a turbine. The flow has a straight sonic line, a monotonic velocity distribution along the body, and uniform supersonic flow at the outflow,

3. The profiling of a lifting supercritical airfoil, with high subsonic speeds and no supersonic embedded regions.

High Acccuracy Compact Upwind Schemes for Viscous Flows

A.I. Tolstykh

USSR Academy of Sciences Computing Center
Vavilova Str. 40, 117 967 Moscow, USSR

ABSTRACT

A general description of third order implicit schemes, based on compact upwind differencing (CUD-3), is given. The main features of these schemes are a three-point stencil, small truncation errors and considerable stability and damping of spurious oscillations. Variations of CUD-3 operators, including their non-conservative, conservative and entropy-consistent forms, are also discussed. Numerical examples are presented to illustrate the behavior of solutions in the presence of discontinuities and/or steep gradients.

Computer codes based on CUD-3 have been developed for a wide range of fluid dynamics problems including laminar and turbulent compressible and incompressible flows. Examples of numerical simulation of external viscous gas flows with separation (in particular, with shock wave-boundary layer interaction) are presented for the supersonic and transonic regimes. Other examples include laminar and turbulent flows in Laval nozzles and in hypersonic wind tunnels. Supersonic flows in channels are also considered.

A brief description of a new family of compact upwind schemes is presented. These schemes are based on fifth order compact upwind differencing (CUD-5), shown to be highly accurate.

Numerical Simulation of Unsteady and Non-Equilibrium Supersonic Flows

L.I. Turchak

Association of Computational Fluid Dynamics
Vavilov Street 40, 117 967 Moscow, USSR

ABSTRACT

Inviscid supersonic flow over bodies is considered. Numerical simulation is carried out by the splitting method into physical processes and by the grid-characteristic method.

Supersonic non-equilibrium blunt body problems are solved for the following gas mixtures:

1. the Earth's atmosphere, with vibrational relaxation and dissociation;
2. carbonic acid-nitrogen-argon, including the Venus atmosphere, with chemical reactions;
3. hydrogen-oxygen, with an exothermal reaction (combination).

The results are obtained for a wide set of the initial parameters, from frozen flows to equilibrium ones.

Unsteady gas dynamics problems are considered for the following:

1. longitudinal oscillations of incoming stream or a body;
2. fast decceleration of moving bodies;
3. flows near bodies with a variable shape, including vibrations;
4. shock interactions near moving bodies.

Comparison of the results with experimental data is made and demonstrates the high efficiency of the numerical method used.

North American, European and Japanese Abstracts

Recent Advances in Computational Aerodynamics

Ramesh K. Agarwal

Program Director CFD
and
McDonnell Douglas Corporation Fellow
McDonnell Douglas Research Laboratories
St. Louis, Missouri 63166-0516, USA

ABSTRACT

The current state-of-the-art in computational aerodynamics is described. Recent advances in the discretization of surface geometry, grid generation, and flow simulation algorithms have led to flow field predictions for increasingly complex and realistic configurations. As a result, computational aerodynamics is emerging as a crucial enabling technology for the development and design of flight vehicles.

Examples illustrating the current capability for the prediction of aircraft, launch vehicle and helicopter flow fields are presented. Unfortunately, accurate modeling of turbulence remains a major difficulty in the analysis of viscous-dominated flows.

In the future, inverse design methods, multi-disciplinary design optimization methods, artificial intelligence technology and massively parallel computer technology will be incorporated into computational aerodynamics, opening up greater opportunities for improved product design at substantially reduced costs.

Co-Located Equal-Order Primitive Variables Formulations in Control Volume Finite Element Methods

B. Rabi Baliga

Department of Mechanical Engineering
McGill University
817 Sherbrooke St. West, Montreal, Quebec H3A 2K6, Canada

ABSTRACT

Control-volume finite element methods (CVFEMs) combine key ideas of finite element and finite volume methods. Thus they provide the geometric flexibility traditionally associated with finite element methods. In addition, they have features characteristics of finite volume methods: Their formulation is amenable to easy physical interpretation, and their solutions satisfy local and global conservation requirements, even for coarse grids.

The formulation of CVFEMs typically involves the following basic steps:

(1) discretization of the calculation domain into elements;
(2) a further discretization of the domain into control volumes that surround the nodes in the finite element mesh;
(3) prescription of element - based interpolation, overshape, functions for the dependent variables;
(4) use of the subdomain, or control volume, form of the method of weighted residuals to derive algebraic equations.

In numerical solutions using the primitive variables model of incompressible viscous fluid flow, if the velocity components and pressure are stored at the same nodes and interpolated using similar functions, physically unrealistic pressure harmonics can arise. Special procedures are required to circumvent or overcome these difficulties. In this presentation, some recent advances pertaining to co-located, equal order, primitive variables formulations in CVFEMs for incompressible viscous flows will be discussed.

Viscous-Inviscid Interaction Analysis of Compressor Cascade Performance

Mark Barnett

Computational Sciences Department
United Technology Research Laboratories
Silver Lane, East Hartford, Connecticut 06108, USA

ABSTRACT

A viscous-inviscid interaction technique for the analysis of compressor cascade blade rows has recently been developed and demonstrated for the prediction of performance of two-dimensional, transonic cascades over the entire incidence range. The analysis has been used to investigate a series of compressor cascade viscous flows, for which corresponding experimental data and Navier-Stokes solutions are available.

The present results show that the viscous-inviscid interaction procedure is able to accurately predict cascade losses and surface pressure distributions. In addition, large scale flow separations spanning more than 80% of blade chord have been calculated, and the predictions are in good agreement with those obtained using a Navier-Stokes analysis.

The results from the present numerical investigation of the form of cascade loss, exit gas angle and pressure distributions have been compared with both experimental data and the Navier-Stokes solutions.

A brief discussion of ongoing efforts to develop an interaction analysis using a potential flow inviscid model will also be given.

On the Finite Element Solution of Incompressible Fluid Flows with Heat Transfer

Klaus-Jürgen Bathe

Department of Mechanical Engineering
Massachusetts Institute of Technology
77 Massachusetts Avenue
Cambridge, Massachusetts 02139, USA

Mo-Hwa Wang and Jian Dong

ADINA R&D, Inc.
Watertown, Massachusetts 02172, USA

ABSTRACT

The objective in this presentation is to give an overview of some experiences recently gathered in the application of finite element procedures to fluid flows.

The finite element discretization and iterative solution schemes used are reviewed, and emphasis is then directed to some valuable experiences obtained with the solution schemes.

The primitive variables form of the Navier-Stokes equations is used with the Galerkin method. The finite elements employed satisfy the inf-sup condition of Brezzi and Babuska. Stabilization for higher Reynolds and Peclet numbers is achieved by a least-squares operator or element bubble functions. The finite element equations are solved using Newton-Raphson iteration with line searching, using either a direct skyline column reduction solver or biconjugate gradient and GMRES iterative solvers.

The experiences to be presented relate to the use of appropriate meshes, the stabilization techniques, the imposition of the boundary conditions, and the use of iterative solvers.

Adaptive and Moving Meshes

Ricardo Camarero

Département de mathématiques appliquées
École Polytechnique de Montréal
C.P. 6079, succursale A, Montréal, Québec H3C 3A7, Canada

ABSTRACT

A current trend of CFD methods is the intimate coupling of the flow solver and grid. This is an especially interesting concept for the computation of unsteady compressible flows, with moving boundaries and moving discontinuities. In these cases, the numerical solution requires a sequence of meshes to accurately describe the evolution of the flow and of the computational domain. Moreover, a numerical scheme which correctly takes into account the grid movement is also needed. In this work we address these aspects and a methodology is presented for the time dependent grid management of transient flow simulations using a Lagrangian-Eulerian formulation of Roe's Euler solver for 2-D/axisymmetric Euler equations.

When transient flow simulations are undertaken, regridding or grid adaptation is necessary for two main reasons. The first is purely geometric. In fact, after moving a grid for a number of time steps, the grid becomes deformed and some equidistribution procedure must be implemented to alleviate this problem. The second reason emanates from the physics as regions of strong gradients continuously move during flow evolution and require a higher grid density in order to capture. To perform these tasks, the subdivision of the domain into triangles has been chosen because of the flexibility of triangulations in local enrichment or coarsening.

Various techniques can be found in the literature for triangular mesh generation and adaptation, most being based on either the frontal method or the Delaunay method. Considering that the goal is the development of a grid management system for the simulation of unsteady flows in complex geometries with moving boundaries, a different approach has been selected with a localized action on the grid at a given time. The meshing system consists in a series of local actions which can refine, coarsen, cure and/or smooth the grid in any given region of the triangulation.

The numerical simulation of unsteady flows using moving grids must respect the so called geometric conversion laws, which establishes relations for the conservation of surfaces and volumes of the control cells. If these laws are violated, a misrepresentation of the convective velocities or extra sources is encountered. According to the intrinsic nature of these laws, they have been formulated using a geometric rationale. This simple, but fundamental idea, denoted by IGCL is based on the evaluation of the volumetric changes along elementary faces of a triangle as it moves.

A Spectral Element Method Applied
to
Parabolized Simulations of Stability and Transition

Warren H. Finlay

Department of Mechanical Engineering
University of Alberta
Edmonton, Alberta T6G 2G8, Canada

ABSTRACT

A spectral element method is discussed that solves the incompressible, spatially-parabolized Navier-Stokes equations with the novel feature that time-dependence is allowed. The method combines the high spatial accuracy of spectral methods with the geometric flexibility of finite element methods.

The code was developed to explore stability and transition in three dimensional, spatially developing flows, without the prohibitive computing requirements and inflow/outflow boundary condition difficulties associated with a fully elliptic spatial description. The restriction of temporal steadiness associated with previous finite difference parabolized methods is removed by using a Fourier decomposition in time. Spatially, the shape functions in each spectral element are high order Legendre polynomials, obtained as the Lagrange interpolants at the nodes of the Gauss-Lobatto quadrature (for velocity) or Gauss quadrature (for pressure) relative to the Legendre weight function. The nonlinear convection terms are marched downstream using a third order Adams-Bashforth method, while the linear viscous diffusion terms are advanced downstream using implicit Euler. The incompressibility constraint is satisfied using a saddle point decomposition of the weak variational formulation.

The successful application of the method to examine the stability and transition of vortices in various flows is discussed.

A Posteriori Error Estimators in Compressible Viscous Flows Simulation

Michel Fortin

Département de mathématiques et de statistiques
Université Laval, Ste-Foy, Québec G1K 7P4, Canada

ABSTRACT

Error estimation is a central issue for the application of numerical simulation to actual engineering situations. Mesh adaptation also requires a good local error estimator. We present an idea of André Fortin which uses two different closely related elements , with bubble functions, to estimate local error levels. We also extend this idea to compressible flow using a SUPG approximation with two different values of the error parameter. The equivalence of the bubble functions approach and SUPG is discussed from results of Brezzi, Franca and Marini.

Control of Artificial Viscosity Effects in Flow Simulations

Mohamed M. Hafez

Department of Mechanical Engineering
University of California, Davis, California 95616, USA

ABSTRACT

Explicit or implicit artificial viscosity is needed in flow simulation for numerical stability and for capturing shocks and contact discontinuities.

In many calculations, artificial vorticity is generated due to the artificial viscosity and the numerical treatment of boundary conditions. Therefore, in aerodynamic calculations, it is desirable to minimize the effect of artificial viscosity by choosing the form and magnitude of the artificial viscosity as well as by a proper treatment of boundary conditions.

Examples of controlling the artificial viscosity effects will be discussed for inviscid and viscous, incompressible and compressible flow calculations.

A Finite Element Navier-Stokes Algorithm Using Fully-Coupled Direct and Iterative Solvers

Wagdi G. Habashi

Department of Mechanical Engineering
Concordia University, Montreal, Quebec H3G 1M8, Canada
and
Aerodynamics Consultant, Pratt & Whitney Canada
1000 Marie Victorin, Longueuil, Quebec J4G 1A1, Canada

ABSTRACT

This paper presents a Finite Element solution method for the incompressible and compressible Navier-Stokes equations, in primitive variables form. To provide the necessary coupling between continuity and momentum, and enhance stability, a pressure dissipation in the form of a Laplacian is introduced into the continuity equation. The governing equations are discretized by a Galerkin weighted residual method and, because of the modification to the continuity equation, equal interpolation for all the unknowns is permitted. Newton linearization is used and, at each iteration, the linear algebraic system is solved in a fully-coupled manner by direct or iterative solvers.

For direct methods, a vector-parallel Gauss elimination method is developed that achieves execution rates exceeding 2.3 Gigaflops, i.e. over 86% of a Cray YMP-8 current peak performance. For iterative methods, preconditioned conjugate gradient-like methods are studied and good performances, competitive with direct solvers, are achieved. Convergence of such methods being sensitive to preconditioning, a hybrid dissipation method is proposed, with the preconditioner having an artificial dissipation that is gradually lowered, but frozen at a level higher than the dissipation introduced into the physical equations.

Convergence of the Newton-Galerkin algorithm is very rapid. Results are demonstrated for two-and three-dimensional incompressible and compressible flows.

Multi-Dimensional Essentially Non-Oscillatory (ENO) Schemes for General Geometries

Ami Harten

ICASE, NASA-Langley Research Center
Hampton, Virginia 23665, USA

ABSTRACT

In this talk we present a class of ENO schemes for the numerical solution of multi-dimensional hyperbolic systems of conservation laws on structured and unstructured grids. This is a class of shock capturing schemes which are designed to compute cell-averages to high order of accuracy. The ENO scheme is composed of a piecewise polynomial reconstruction of the solution from its given cell-averages, approximate evolution of the resulting initial value problem, and averaging of this approximate solution over each cell. The reconstruction algorithm is based on an adaptive selection of a stencil for each cell so as to avoid spurious oscillations near discontinuities, while achieving high order of accuracy away from them.

The work was carried out in collaboration with Dr. Sukumar R. Chakravarthy, Rockwell Science Center, Thousand Oaks, California.

A New Lagrangian Approach for Solving the Euler Equations

Grafton W.H. Hui

Department of Applied Mathematics
The University of Waterloo
Waterloo, Ontario N2L 3G1, Canada

ABSTRACT

Most existing shock capturing methods for solving the multi-dimensional, steady, Euler equations of gas dynamics are based on the Eulerian method of flow description. They resolve flow discontinuities, and slip lines in particular, rather poorly.

A generalized Lagrangian method has recently been developed. For 2-D steady flow the independent variables are a stream function and the distance travelled by a particle along its streamline.

Extensive comparisons with exact analytic solutions, and with experiments, show that the generalized Lagrangian method is superior to the Eulerian one in the following aspects:

- it resolves the slip line discontinuity crisply, whereas Eulerian methods deteriorate with increasing slip line strength,
- its shock resolution improves greatly with increasing Mach number, and
- it requires no grid generation.

Algorithms for Flow Optimization and Control

Max D. Gunzburger

Department of Mathematics
and
Interdisciplinary Center for Applied Mathematics
Virginia Polytechnic Institute & State University
Blacksburg, Virginia 24061-0123, USA

ABSTRACT

The control or optimization of fluid motions for the purpose of achieving some desired objective is crucial to many technological applications. In the past, such control problems have been addressed, for the most part, either through expensive experimental processes or through the introduction of significant simplifications into the development of control mechanisms and approximation algorithms. Only recently have flow control and optimization problems been addressed by engineers and mathematicians in a systematic manner, without invoking unrealistic assumptions. This interest is quickly expanding so that at this time, flow control and optimization is rapidly becoming a very active and successful area of inquiry.

We describe some of the recent progress made with regard to the approximate solution of flow control and optimization problems. We begin by discussing various possible objectives (e.g. drag minimization, flow conformation, enhancing or deterring mixing, avoiding hot spots) of such problems. We then discuss some of the possible control mechanisms (e.g. injection or suction of fluid, heating or cooling, shape modification) that can be used to achieve the desired objective. Finally, we discuss some methods for finding approximations of flow control and optimization problems. Included in our discussion are adjoint (or co-state or Lagrange multiplier) methods and methods based on the direct computation of sensitivities, i.e. of the derivatives of the state with respect to the control parameters. We close by presenting the results of some numerical experiments.

Non-Equilibrium Viscous Shock-Layer Calculations of Glassy Ablators

Walter L. Melnik

Aerospace Engineering Department
The University of Maryland
College Park, Maryland 20742, USA

ABSTRACT

Validation of computer programs used in the calculation of glassy ablators has often been based upon comparisons with experimental results for artificial tektite glass models obtained in a supersonic arc jet. The estimates of aerodynamic heating were based on correlations of measurements on a surface which catalyzed the recombination of oxygen or nitrogen atoms, whereas tektite glass is essentially inert to such reactions. Consequently the estimated aerodynamic heat transfer rates were higher than those actually encountered by the tektites. Perhaps the most puzzling aspect was the apparently good agreement between previous calculations and the experimental results obtained on ablation in a supersonic arc jet by Chapman and co-workers. The amount of ablation, together with the wavelength of concentric rings imprinted on the melt layer when it solidified, would normally serve to define the entry conditions of tektite trajectories. Previous studies of tektite ablation supported a lunar origin in contrast to the geochemical argument that tektites must be the result of meteorite impact on earth. The previous results would, at first glance, appear to understate the case for a lunar origin.

Starting from prescribed entry conditions, the trajectories of initially spherical tektites were obtained from numerical integration of Newton's second law using a fourth-order Runge-Kutta method. The rate of ablation, shape and instantaneous mass of the tektite were obtained from numerical solution of the governing equations, using appropriate estimates of the aerodynamic heating. A separate assessment using thin-layer Navier-Stokes solutions was made of the aerodynamic heat transfer rates for representative flight conditions. In particular, the flow conditions of Chapman's supersonic arc-jet experiments were simulated, with the free stream consisting of fully dissociated oxygen and partially dissociated nitrogen. Parametric studies of atmospheric reentry conditions were carried out to achieve matching in the ablation depth of tektites found in Australia. Trajectories representative of a lunar origin were modeled using a parabolic entry speed. For the terrestrial scenario, tektites were presumed to originate from entrainment of splash ejecta by vapor cloud produced by impact. It was assumed that they were launched into space along ballistic (suborbital) trajectories, whereupon they cooled by radiation and solidified prior to reentry to earth's atmosphere.

Application of Navier-Stokes Methods to Rotorcraft Aerodynamic Design and Analysis

Jim C. Narramore

Manager, Computational Fluid Dynamics
Bell Helicopter TEXTRON, Inc.
Fort Worth, Texas 76101-0482, USA

ABSTRACT

Advancements in methodology for the practical simulation of flowfields about complex configurations using numerical solution of the equations of fluid motion have been dramatic in recent years. The tremendous potential of this relatively new technology has been demonstrated in the fixed-wing aircraft industry, where complete airplane configurations are now being modeled by aerodynamicists using Navier-Stokes computational fluid dynamic methods. However, in the rotorcraft industry, Navier-Stokes technology is just beginning to be studied and extended for application to helicopter and tilt-rotor configurations. At the present time, components of the total rotorcraft vehicle are being modeled using Navier-Stokes methods. Airfoil, wing, and rotor blade configurations have been investigated to date.

The main focus of this presentation will be to reflect on design applications of CFD within the environment of Bell Helicopter Textron Incorporated (BHTI). It is basically a snapshot of some of the things that may be done today using Navier-Stokes codes, and the list of cases presented is neither complete nor comprehensive. However, it does show the types of problems that are currently being solved.

The presentation will include results from a:

- 2-D Navier-Stokes inverse method that determines the airfoil geometry required to produce a specified pressure distribution
- 2-D multi-element Navier-Stokes analysis code to compute the flow about wing-flap-slat configurations
- 3-D Navier-Stokes wing code that has been used to investigate tip effects on flow characteristics
- 3-D Navier-Stokes rotor code that has been used to investigate stall distributions on rotorcraft blades.

Complementary Volume Schemes in Incompressible Flow

Roy A. Nicolaides

Department of Mathematics
Carnegie Mellon University
Pittsburgh, Pennsylvania 15213, USA

ABSTRACT

Complementary volume schemes form a natural generalization to unstructured meshes of the classical staggered mesh MAC schemes. However, there is much more flexibility for obtaining the implicit velocity components. In this presentation, the basis for the generalizations will be presented and the main results obtained so far will be discussed.

Recent CFD Research and Development in Asia-West Pacific Area

Koichi Oshima

Japan Society of Computational Fluid Dynamics
2-7-7 Sekimachi-kita, Nerima-ku, Tokyo 177, Japan

ABSTRACT

The following CFD meetings, ICNMFD (International Conference on Numerical Methods in Fluid Dynamics) and ISCFD (International Symposium on Computational Fluid Dynamics) have taken place or will be held:

1985	Tokyo	1st ISCFD	1986	Beijing	10th ICNMFD
1987	Sydney	2nd ISCFD	1988	Williamsburg	11th ICNMFD
1989	Nagoya	3rd ISCFD	1990	Oxford	12th ICNMFD
1991	Davis	4th ISCFD	1992	Rome	13th ICNMFD

In addition the following Workshops took place:

1988	Khabarovsk:	1st Soviet Union-Japan CFD Workshop
1990	Tsukuba:	2nd Soviet Union-Japan CFD Workshop

The papers presented at these meetings have been published, in various forms, after refereeing and editing.

The number of papers presented at the conferences, and the attendance, have been growing steadily, because of the remarkable progress in this field of science. Interest from the Asia-Pacific area has been particularly growing. ICNMFD-Beijing (1986) was perhaps the first such occasion we had in Asia, while ISCFD-Tokyo (1985) was the first instance in which a large group of Soviet scientists interacted with western scientists. Both events have given strong impetus to CFD in the Asia-West Pacific area.

Soviets have traditionally been very strong in mathematics and mechanics while also keeping a high profile in fundamental aspects of CFD; Chinese researchers have been known to be quite gifted in science and play a major role in CFD, inside and outside China. Japanese made supercomputers are now dominating the world, and, as a matter of fact, are enjoying 70% of the world market. Japanese CFD scientists are now allowed "unlimited" CPU time and are not limited by resources. Thus, Soviet mathematics, Chinese human scientific power and Japanese computers is an equation that will make the progress of this CFD research and development explosive.

Modeling of Drag Reduction by Passive Means

Andrew Pollard

Department of Mechanical Engineering
Queen's University
Kingston, Ontario K7L 3N6, Canada

ABSTRACT

Drag reduction by passive means can be achieved in a number of ways; however, in this paper the use of near-wall manipulators and riblets will be highlighted. Manipulators are thin aerofoil shaped blades that are placed in proximity to a wall, the wake from which results in a reduction in wall skin friction from that in a plain boundary layer. Riblets achieve drag reduction, it is thought, by hampering the lateral spreading of near wall turbulence structures. The modelling of these two phenomena are considered by way of example: manipulated pipe flow and ribleted boundary layer flow.

Two-dimensional turbulent flow inside pipes is calculated with and without the addition of single manipulators or tandem manipulators. In the latter case, the effect of inter-manipulator spacing is considered. These calculations are obtained using a two-equation low Reynolds number model of turbulence with over 30,000 control volumes required to adequately resolve the flow. The main result is that, when compared to experimental data, good agreement is obtained. However, even though the data indicates that drag reduction may be achievable, the calculations do not confirm this finding.

The flow over a ribleted wall is considered. A review of the work to date will be presented including the modelling achievements of other workers. The simulation strategy considered by the author and his co-workers will be outlined and preliminary results will be presented.

Element-by-Element Methods for the Solution of Incompressible Three-Dimensional Flows

J.N. Reddy and M.P. Reddy[1]
Department of Engineering Science & Mechanics
Virginia Polytechnic Institute & State University
Blacksburg, Virginia 24061, USA

ABSTRACT

Finite element analyses of complex flow problems of practical interest result in the solution of a large system of algebraic equations in terms of the nodal values. Solution of equations is the most time-consuming part of the analysis and accounts for nearly 80% of the CPU time and storage. The equations can be solved either by using direct elimination methods or by iterative methods. Direct solvers rely on the elimination techniques and provide the solution after a fixed number of steps. Generally, they are insensitive to the conditioning of the coefficient matrix associated with the equations. However, the main deficiency of the direct solvers when used to solve large system of equations is that they require the matrix to be stored in an ordered format, resulting in prohibitively excessive storage and cumbersome bookkeeping. Elimination methods are also computationally intensive. The limitations on CPU time and storage requirements therefore make the use of direct solvers inefficient and impractical even on the present day high speed and large memory supercomputers.

These limitations on storage can be overcome by the element-by-element data structure, where information is stored and maintained at the element level rather than assembled into a global data structure. The advantages of the element-by-element data structure over direct solvers are: (i) zeroes are not stored, hence the total storage and computational costs are low; (ii) the amount of storage is independent of the node numbering, and mesh topology and depends on the number and type of elements in the mesh; and (iii) the algorithms can be vectorized for efficient use on supercomputers. Further, the element-by-element data structure is natural to finite element analysis on high-speed, parallel processor supercomputers.

Element-by-element algorithms have been used to solve a variety of engineering problems but not incompressible fluid flow, especially the penalty function formulation. The present work was undertaken to study the computational efficiency of 3 iterative schemes, (i) Bi-orthogonal Conjugate Gradient, (ii) Lanczos ORTHORES, and (iii) GMRES for the analysis of viscous, incompressible flows in three dimensions. A number of numerical examples governed by coupled fluid flow and heat transfer equations will be discussed in terms of the accuracy and computational cost of each scheme.

---

[1] presently at: Technalysis, Indianapolis, IN 46268

The Role of CFD in Airplane Design

Paul E. Rubbert

Unit Chief, Aerodynamics Research
Boeing Commercial Airplane Group
Mail Stop 7H-91, Seattle, Washington 98124-3707, USA

ABSTRACT

The underlying design objective in the commercial jet transport business is to design and produce an airplane that will provide the best economic return in its chosen market, with competitive or superior levels of passenger comfort, etc. That goal can be acheived only through the use of design processes that offer the ability to achieve the right balance between aerodynamic efficiency, structural design and weight, and airplane initial cost and in - service maintenance costs.

It is now becoming clear that the impact of CFD on airplane design goes far beyond the ability to achieve superior aerodynamic performance. Another major contribution of CFD is to permit a shortening of the aerodynamic design process, a shortening that is key to doing a better job of engineering the trades between aerodynamic shape, structural weight, and cost to build and maintain. And that is key in arriving at an overall design that will produce the best economic return for the airlines.

The talk discusses the growing role of CFD in contributing to improved airplane economic performance and gives a number of examples wherein the economic performance of various Boeing airplanes have been improved through contributions of CFD.

Advances in Control Volume-Based Finite Element Methods for Compressible Flows

Gerry E. Schneider

Department of Mechanical Engineering
The University of Waterloo
Waterloo, Ontario N2L 3G1, Canada

ABSTRACT

Recent developments in the use of control volume-based finite element methods for the prediction of incompressible flows have been highly successful. Most notably, the use of integration point operators for the determination of convected flow variables has solved the pressure checkerboard problem as well as considerably enhanced the prediction accuracy.

The extension of these concepts to compressible fluid flow prediction is described in the present paper, as well as the performance of these procedures. In addition, the use of the Second Law of Thermodynamics in the determination of the required amount of artificial dissipation is described in a control volume-based context. The combination of integration point operators with Second Law dissipation is shown to yield highly accurate solutions in the application to three test problems.

Computational Aerodynamics in Germany

W. Schmidt and E.H. Hirschel

Director Air Vehicle Engineering
and
Manager, CFD Group
Messerschmitt-Bölkow-Blohm, GmbH
Postfach 801160, 8000 München 80, Germany

ABSTRACT

1. Major computational aerodynamics groups in Germany

2. German aerospace projects and products

3. Selected CFD results

4. Conclusions

Impact of Supercomputing in the Aerospace Sciences

Vijaya Shankar

Director, Computational Sciences
Rockwell International Science Center
Thousand Oaks, California 91360, USA

ABSTRACT

The development of modern aerospace vehicles increasingly requires synergism in integrating multi-disciplinary technologies such as:

- fluid dynamics for flow management,
- structures for flexibility effects,
- propulsion for thrust,
- controls for stability,
- low observables for stealth considerations.

Implementing such an integrated approach demands development of a computational environment referred to as "Computational Sciences", that encompasses many disciplines. The mission of Computational Sciences is to combine the strengths of mathematics and supercomputers to better understand/simulate problems in physical sciences and integrate such multidisciplinary technologies to achieve synergism in the design process.

One of the computational disciplines, Computational Fluid Dynamics (CFD), has been the cornerstone for innovative developments in numerical algorithms to study many different types of problems in fluid physics by using a hierarchy of equations of varying complexity, most notably the Navier-Stokes equations. The objective of this presentation is to briefly describe the progression of supercomputing, from the CFD point of view, in the establishment of "Computational Sciences" to address multidisciplinary problems impacting the design of advanced aerospace configurations.